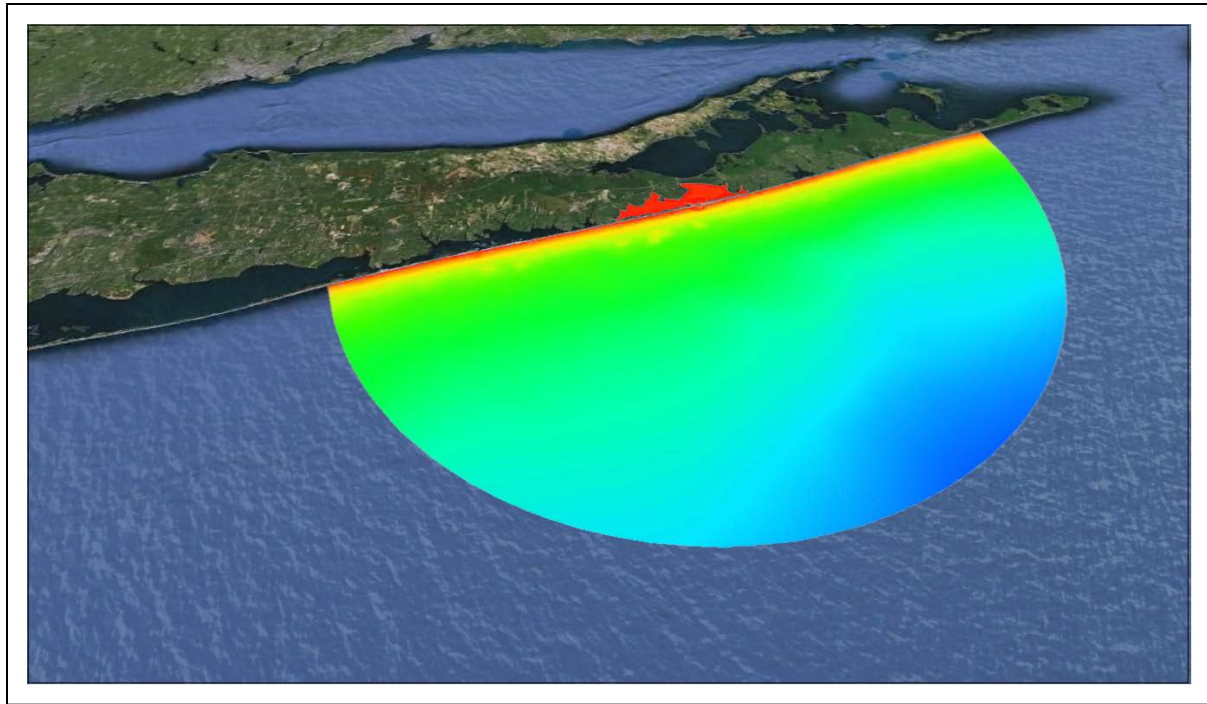


SMS 11.1 Tutorial

ADCIRC Analysis



Objectives

This lesson will teach you how to prepare a mesh for analysis and run a solution for *ADCIRC*. It will cover preparation of the necessary input files for the *ADCIRC* circulation model and visualization of the output. You will start by reading in a coastline file and then a SHOALS file.

The data used for this tutorial are from Shinnecock Bay off of Long Island in New York. All files for this tutorial are found in ADCIRC Data Files directory.

Prerequisites

- Overview Tutorial

Requirements

- ADCIRC
- Map Module
- Mesh Module
- Scatter Module
- LeProvost Tidal Database

Time

- 60-90 minutes

AQUAVEO™



1 Reading in a Coastline File




For this tutorial, you will first read in a coastline file, which has already been set up for you. This sample coastline will form the boundary for your mesh. To set up your coverage for *ADCIRC* and open the coastline file:

1. Change the coverage type to *ADCIRC* by right clicking on *default coverage*, selecting *Type*, and choosing *ADCIRC* (in the models subcategory).
2. Select *File* | *Open*.
3. Select the file *shin.cst* in the Data Files Folder for this tutorial and click the *Open* button.

Coastline files include lists of two-dimensional polylines that may be closed or open. The open polylines are converted to *Feature Arcs* and are interpreted as open sections of coastline. Closed polylines are converted to arcs and are assigned the attributes of islands.

1.1 Defining the Domain


We need to assign a boundary type to the coastline arc, and then we can define the region to be modeled. To do this:

1. Make sure you are in the *Map*  module, if not already selected.
2. Choose the *Select Feature Arc*  tool from the *Toolbox* and click on the coastline arc to select it.
3. Select *Feature Objects* | *Define Domain*.
4. Select the *Semi-circular* option and click *OK*.
5. Frame  the display.

A semi-circular arc is created to define the region.

1.2 Assigning Boundary Types

Boundary types for arcs are specified in the *Map* module. Boundary types are prescribed by setting attributes to Feature Arcs. To set the boundary types:

1. Choose the *Select Feature Arc*  tool from the *Toolbox*.
2. Double click the arc representing the ocean boundary, shown in Figure 1. In the *ADCIRC Arc / Nodestring Attributes* dialog, assign this arc to be of type *Ocean*.

3. Click the *OK* button to close the dialog.

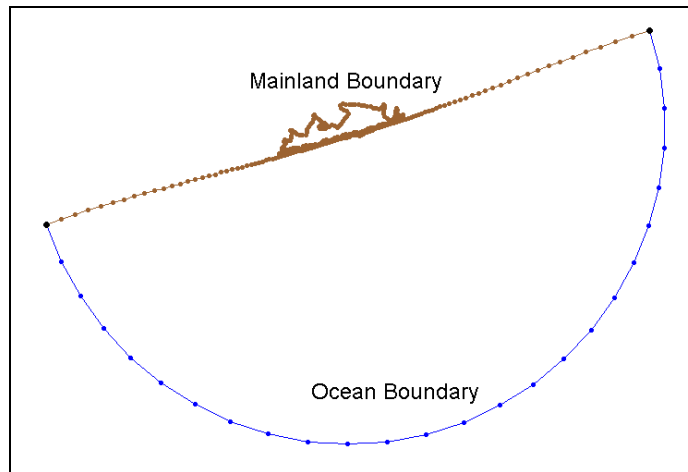


Figure 1 Feature Arcs after boundary types have been assigned.

2 Editing the Coastline File

Now that the coastline file has been read in and a corresponding map object created, several modifications must be made to the data before the SHOALS file is read in. Since the SHOALS file is in UTM coordinates, zone 18(78W to 72 W northern Hemisphere), the data should also be set to those coordinates. In order to do this the display projection must be set to same global projection. To convert the display coordinates:

1. Choose *Edit / Project...*
2. In the *Current Projection* dialog that appears, toggle on *Global projection*. Set the zone to 18 (78W to 72W – Northern Hemisphere) and the datum as NAD 27. Click *OK*.
3. Ensure that the vertical units is set to *Meters*.
4. Click the *OK* button to exit the dialog.

Next, to set the Map object coordinates:

1. Right-click on the "default coverage" map object and select *Projection (floating)*... "floating" indicates that the projection has not been set previously.
2. Repeat steps 2-4 above.

The coastline data has now been converted from a local projection to a global projection of UTM coordinates with zone 18 (78w to 72W--Northern Hemisphere) and Datum set at NAD27.

3 Reading in a SHOALS File

You will now read in a SHOALS file, *shin.pts*, which contains data at various locations along the coastline and throughout the region you are modeling.

1. Choose *File / Open*.
2. Select the file *shin.pts*.
3. In the *Open File Format* dialog, toggle on *Use Import Wizard* and click OK.

The *File Import Wizard* dialog will open, allowing you to specify how the data will be read into *SMS*. For *Step 1* of the dialog, the first line in the *File preview* box is the file header. The next line shows the name of each respective column of data. In this case, the file has three data columns. The first column is the *X* Coordinate, the second column is the *Y* Coordinate, and the third column is the *depth/bathymetry*.

- Click the *Next >* button to move on to *Step 2* of the *File Import Wizard*.

The second step of the *File Import Wizard* allows you to change other specifications as you read in the SHOALS file.

- Click the *Finish* button.

Figure 2 shows the plot of the points read in from the *shin.pts* file. (If the scatter set does not appear at first, make sure that *Points* is toggled on in the display options under *Scatter*).

Once the scatter set is read in, we must set its projection. To do this:

1. Right click on the *shin* scatter set in the project explorer, and click on *projection*.
2. In the *Object Projection* dialog, the projection will already be set to *UTM* since the *Display Projection* is currently set to *UTM*. Click OK to exit the dialog.

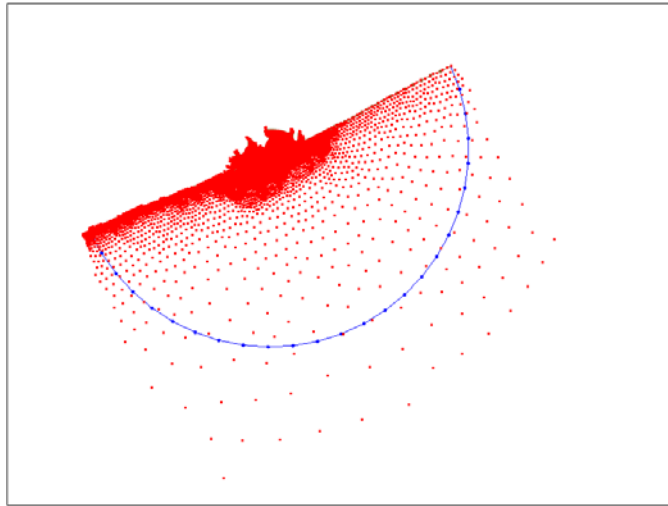



Figure 2 Display of shin.pts.

4 Shallow Wavelength Functions

The next step before you build your finite-element mesh is to create several functions for creating the finite element mesh. For this tutorial, the mesh will be generated according to the wavelength at each node. Large elements will be created in regions of long wavelengths. Conversely, smaller elements are needed closer to the shore to correctly model the smaller wavelengths.

To create this shallow wavelength function from the bathymetric data:

1. In the *Scatter*  module, select *Data / Data Set Toolbox*.
2. In the *Data Set Toolbox* dialog, select the *Wavelength and Celerity* tool in the *Coastal* section. This enables the wavelength options on the dialog.
3. Make sure the options for creating a *Wavelength* and *Celerity* function are checked. Leave the period at 20 seconds and enter a name “20 sec” in the *Output data set names* edit field.
4. Click the *Compute* button to create the data sets and the *Done* button to close the dialog.

Two functions are created: celerity and wavelength at each node using the shallow water wavelength equation. The celerity is calculated as:

- $Celerity = (Gravity * Nodal\ Elevation)^{0.5}$.

The wavelength is calculated as:





- $Wavelength = Period * Celerity$.

5 Creating Size Functions

Now that you have created the wavelength function, you must make a few more conversions before you are ready to create your mesh. A size function is a multiple that guides the size of elements to be created in SMS. Any data set may be used for this purpose. If you were to generate your mesh using the original wavelength function alone, you would get a decent mesh to work with, but we want a mesh whose density radiates out from a point in the inlet. This allows you to get more accurate results in the inlet where we are most concerned with the outcome of the *ADCIRC* run. Therefore, we will now need to create a size function based on the wavelength to attain this end. The final size function you use for modeling applications varies and is found through trial and error to give a nicely formed mesh. This example illustrates one method of building a size function.

5.1 Finding the Central Point for the Mesh

Since the mesh will be generated in a radial fashion, the distance from a central point must be found. The first step is to locate the central point and then use the *Data Calculator* to compute the distances of all points from this center point. To do this:

3. Still in the Scatterpoint  module, zoom in to the area of the inlet shown in Figure 3 with the Zoom  tool until your screen looks like Figure 4.
4. Click on one of scatter points in the middle of the inlet using the Select Scatterpoints  tool. Make note of this point's X and Y coordinates in the Edit Window at the top of the screen.
5. Frame the data by clicking the Frame  tool in the Toolbox.

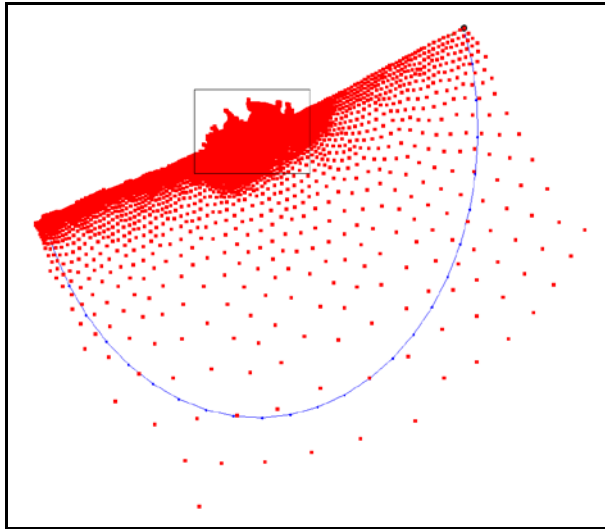


Figure 3 Inlet location to zoom in on.

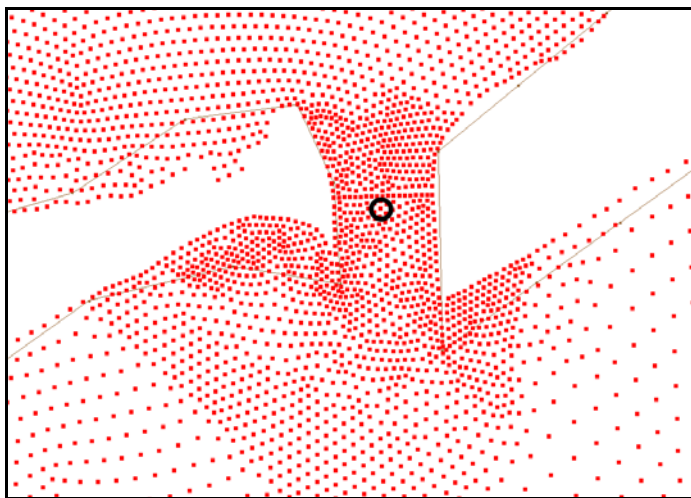



Figure 4 Choose a center point.

For now, turn off the scatterpoint display. However, you may turn it back on at any time during the tutorial if you so desire. To turn off the visibility of the *shin.pts* data unselect the box next to the  *shin* dataset in the *Project Explorer*.

You are now ready to proceed. We will use the *Data Calculator* to compute new data sets by performing operations with scalar values and existing data sets. The *Data Calculator* will be used to create the size function.

5.2 Distance Function

For consistency, we will use the (x,y) location of (712768.675, 4523969.712) as the center scatterpoint for our mesh.

1. Select *Data | Data Calculator...*. This brings up the *Dataset Toolbox*, with the data calculator active.
2. Click the \sqrt{x} button.
3. In the Expression field, using the keyboard replace “?” so the expression looks like:
 - $\text{sqrt}((d4 - 712768)^2 + (d5 - 4523950)^2)$

This expression takes the *x* and *y locations* of each scatter point, which correspond to the “d4” and “d5” data sets respectively, and computes its distance to the point designated as the mesh center.

4. In the *Output data set name* field, enter the name of “distance” for the data set and click the *Compute* button.

5.3 Initial Size Function

1. Highlight the “20 sec_Wavelength” data set and click the *Add to Expression* button. You now should see the letter and number “d2” in the *Expression* field.
2. In the *Expression* field, make the equation look like “d2*7”.
3. Enter the name “size” for this data set in the *Output dataset name* area and click the *Compute* button. This creates a function of 7 times the wavelength.

5.4 Scale Function

The last separate function before computing the final size function will be a scale factor out from the center point. It will take on the following format:

- $\text{scale} = (\text{distance}/\text{max distance})^{0.5}$.

This scale function will range between 0 and 1, 0 being at the center point and 1 at the farthest point from the center of the mesh. This will allow the mesh to radiate out in density from the middle of the inlet. Taking the square root of the scale factor forces the elements to grow larger more quickly as one moves away from the center. To compute this function:

1. Highlight the “distance” function in the Data Sets window and click the Data Set Info... button. Notice that the Maximum value is 65607.9.
2. Click the X in the corner of the dialog window to close the info dialog.

3. Enter “sqrt(d6 / 65607.9)” in the Expression field. This assumes that d6 is the distance function.
4. Enter the name “scale” and click the Compute button.

5.5 Final Size Function

You are now ready to create the final size function that your mesh will be based on.

1. Click the max(x,y) button.
2. Replace “??,??” so the equation reads “max(50, (d7*d8))”. This will multiply the scale factor (which varies from very small by the center of the domain up to one at the edges) by the size (which is seven times that of the wavelength). The result will be a value that varies from very small to seven times that of the wavelength that is truncated to a minimum size of 50 meters to prevent infinitely small elements from being created around the mesh center.
3. Enter the name “radial size” in the Result field and click the Compute button.

The data calculator gives you many options for building the size function. The size function created in this tutorial was created through several steps. This was done to show the many possibilities that exist for defining the size function, and ultimately for defining the finite element mesh. Other options that could be used for this or other meshes include:

- Use the wavelength multiplied by a scale factor (without using distance).
- Don’t take the square root of the scale factor for a denser mesh.
- Use a value other than 50 meters as the minimum size for a denser mesh in the channel, etc.

5.6 Smooth Size Function

The final step in creating a size function is to smooth the size function. Smoothing modifies the size function so the size function values do not change too quickly. Size functions that change too quickly can create poor transitions in element size.

1. In the *Dataset Toolbox*, select *Smooth data sets* from the tool list on the left..
2. Select the scatter data set named “radial size”.
3. Change the Element area change limit to 0.5. This will modify the size function so the elements created by the size function are at most twice as big or half as small as their adjacent elements.
4. Enter the name “radial size smoothed 0.5” and click the Compute button.
5. Click *Done* to exit the *Dataset Toolbox*.


(Note: If desired, the differences between the data set “radial size” and “radial size smoothed 0.5” can be visualized by using the data calculator to subtract “radial size” from “radial size smoothed 0.5” and contouring the resulting data set.)

6 Creating Polygons

A *polygon* is defined by a closed loop of Feature Arcs and can consist of a single Feature Arc or multiple Feature Arcs, as long as a closed loop is formed. For initial mesh generation, polygons are a means for defining the mesh domain.


6.1 Building Polygons

To create polygons from the arcs on the screen:

1. Switch to the *Map*  module.
2. Make sure that no arcs are currently selected.
3. Select *Feature Objects / Build Polygons*. Now a polygon has been created out of all the arcs.

6.2 Polygon Attributes

Next, each polygon (this case only has one) must be assigned proper attributes.

1. Choose the *Select Feature Polygon*  tool from the *Toolbox* and click inside the polygon.
2. Select *Feature Objects / Attributes*. (Double-clicking inside the polygon will perform this same step.) The *2D Mesh Polygon Properties* dialog will open.

6.3 Assigning the Meshing Type

1. Select *Scalar Paving Density* as the Mesh Type.
2. Click the *Scatter Options...* button below the Mesh Type.
3. In the *Interpolation* dialog, in the dataset tree under *Scatter Set To Interpolate From*, make sure the “radial size smoothed 0.5” function is highlighted.
4. In the Extrapolation section, set the Single Value to 50.
5. Turn on the *Truncate values* option and set the *Min* to 50 and the *Max* to 5000.
6. Click the OK button to return to the 2D Mesh Polygon Properties dialog.

6.4 Assigning the Bathymetry Type

Next, the bathymetry type is selected. In this case the imported bathymetry is in the form of a scatter set.

1. Select *Scatter Set* as the *Bathymetry Type*.
2. Click the *Scatter Options...* button below the *Bathymetry Type* option.
3. In the *Interpolation* dialog, highlight *Z* under *Scatter Set To Interpolate From*, leave the *Single Value* at 0.000, and make sure the *Truncate values* option is turned off.
4. Click the *OK* button.

6.5 Assigning the Polygon Type

1. Make sure the Polygon Type is set to Ocean.
2. Click the OK button to close the 2D Mesh Polygon Properties dialog.



7 Creating the Mesh

Once the polygon attributes are set, the mesh can be generated automatically based on the options that were selected. To generate the mesh:

1. Select Feature Objects | Map -> 2D Mesh.
2. Turn off the toggle for *Copy coverage before meshing* and click *OK*.

7.1 Mesh Display Options

After *SMS* has completed generation of the mesh, you should be able to view the bathymetry, nodes, and elements. To set the display:

1. Switch to the Mesh  module.
2. Select Display | Display Options... or select the  macro from the Toolbox.
3. Make sure the Nodes and Contours are turned off and the Elements are turned on.
4. Click the OK button to close the Display Options dialog.

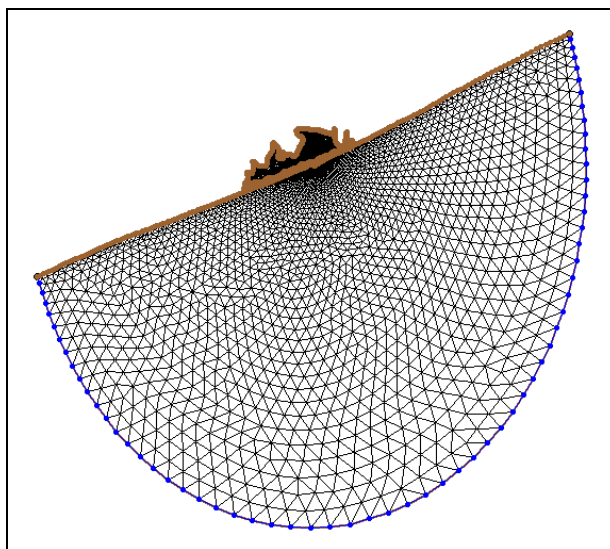



Figure 5 View of elements after automatic mesh generation.

Figure 5 shows the final mesh. Notice how the elements are smaller closer to the coast and within the inlet. Once the mesh has been created and refined, final preparations must be done in order to run *ADCIRC*. These items are renumbering of the mesh nodes and saving the grid.

7.2 Minimizing Mesh Bandwidth

Before running *ADCIRC*, the mesh nodes must be renumbered to minimize the bandwidth of the mesh. This allows the *ADCIRC* model to run efficiently. SMS has done this automatically as the mesh was generated. However, if you edit the mesh, you would need to renumber again. To do this:

1. Select the Select Nodestring  tool from the Toolbox and select the nodestring along the ocean boundary.
2. Select Nodestrings | Renumber Nodestrings.

The nodes have now been renumbered for the entire mesh starting with those along the ocean boundary.

8 Building the *ADCIRC* Control File

The control file specifies values corresponding to different parameters for *ADCIRC* runs. These parameters include specifications for tidal forcing, selection of terms to include,

hot start options, model timing, numerical settings, and output control. In order for *ADCIRC* to run properly, the mesh must be converted to Latitude/Longitude coordinates.

8.1 Converting Back to Lat/Lon

The model control expects the coordinates to be in latitude/longitude. The initial conversion was made to UTM coordinates for the meshing. The size function was calculated in meters, so the mesh could not be created while the coordinates were in degrees without performing more conversions (i.e. degrees \leftrightarrow meters). To convert back to Geographic coordinates:

1. Right click on the mesh object in the Project Explorer.
2. Select *Reproject*. In the *New Projection* side, Click on *Set Projection* and set the projection to *Geographic* and leave the datum as *NAD 27*.
3. Make sure the *Vertical* units are in *Meters*.
4. Click the OK button.
5. To change the *Display Projection* to Geographic lat/lon as well, right click on the mesh object in the Project Explorer and select *Work in Object Projection*. (Note that this is an optional step and does not affect the settings of the model, just our view of the model).

8.2 Main Model Control Screen

Before we set the model control, we need to make sure that SMS points to the right location of the LeProvost tidal database, as SMS will need to access the database at some point in the model control. To do this:

1. Select *Edit / Preferences* and click on the *File Locations* tab.
2. In the *Other Files* section of the dialog, set the *LeProvost tidal database* to point to the location where the LeProvost tidal database files are stored. They may be found in the Data Files folder of this tutorial.

To set up the model control for *ADCIRC*:

1. Select *ADCIRC | Model Control*.
2. In the General tab, turn on the following options under Terms in the center of the dialog: Finite Amplitude Terms, Wetting/Drying, Advective Terms, and Time Derivative Terms.
3. Click the Options... button below the Wetting/Drying option.

4. Make sure the following values are entered in the Wetting/Drying Parameters dialog:
 - *Minimum Water Depth*..... 0.05
 - *Minimum Velocity for Wetting*..... 0.02
5. Click *OK* to return to the *Model Control* dialog.
6. Enter a value of 3.0 for the *Lateral Viscosity* in the *Generalized Properties* section on the right side of the dialog.
7. Change the *Bottom Stress/Friction* method to *Constant Quadratic* and set the *Friction coefficient* to 0.005.

8.3 Time Control

Next, values for the *Timing* must be set. To set these values:

1. Click on the *Timing* tab.
2. Set the following values:
 - *Ramp function value*: 1.0 days. (This is a time period for the model to ramp from no circulation to full tidal amplitude. This enhances stability. The results from these calculations do not match physical reality and are normally not even saved.
 - *Time Step*: 2.0 seconds
 - *Run Time*: 1.5 days (Normal simulations last for several days up to a full lunar month. This is set to 36 hours just to get past the ramp time and show a tidal cycle.)

8.4 Output Files

ADCIRC will generate two global output files, water-surface elevation and velocity. To set the time for the two files:

1. Click on the *Files* tab.
2. In the *Output Files Created by Adcirc* section scroll down to *Elevation Time Series (Global)* and turn the *Output* checkbox on.
3. Make sure the *Start (day)* is set to 1.0 and set the *End (day)* to 1.5 and the *Frequency (min)* to 30.
4. Repeat steps 2 and 3 for the *Velocity Time Series (Global)*.

8.5 Tidal Forces

For this run of *ADCIRC*, tidal forcing will be used. To define the tidal constituents that *ADCIRC* will apply at the ocean boundaries:

1. Click on the *Tidal/Harmonics* tab.
2. Check the *Use forcing constituents* and *Use potential constituents* boxes in the *Tidal Constituents* section.
3. Click the *New* button next to the checkboxes.
4. In the New Constituent dialog, make sure the LeProvost constituent database is selected. Select the K1, M2, N2, O1, and S2 constituents in the Constituents section.
5. Set the Starting Day as 0.0 hours on February 1, 2000 (Hour: 0.0, Day: 1, Month: 2, and Year: 2000). This is the date from which the tides will start.
6. Click the OK button to return to the ADCIRC Model Control dialog.

SMS takes each constituent, extracts the values it needs from the *LeProvost* constituent database, and places them into the spreadsheet in the lower left corner. The amplitude and phase values may then be adjusted for each node.

If a message appears indicating that *SMS* cannot find one of the constituent files, click *OK* and find the file. The file should be located with the tutorial files.

7. Click the OK button to exit the ADCIRC Model Control dialog.

8.6 Saving the Mesh and Control Files

To save the mesh and control files:


1. Select File | Save New Project...
2. Enter the name *shinfinal.sms* and click the Save button.

9 Running *ADCIRC*

You are now ready to run *ADCIRC*. Presently, *ADCIRC* uses a specific naming convention for its input and output files. Therefore, before *ADCIRC* can start, the basic input files must be present in the working directory, which *SMS* takes care of automatically. *SMS* makes a copy of the active mesh file and names it *fort.14*, then makes a copy of the model control information file and names it *fort.15*.

To run *ADCIRC*:

1. Select ADCIRC | Model Control.
2. Select the General Tab. Name your project "test" in the *Project title:* and *Run ID:* "test" as well. Press *OK*.

3. Select ADCIRC | Run ADCIRC.
4. If the name of the ADCIRC executable does not appear, click the folder icon , locate the ADCIRC executable, and click OK.

The *ADCIRC* model wrapper appears and gives status for 64800 timesteps while the model runs. On a typical desktop machine, this will take around 15 minutes. Once the *ADCIRC* run has completed, there will be several new files created. SMS copied the *shinfinal.grd* file (the mesh file saved when the project file was saved) to *fort.14* and *shinfinal.ctl* file to *fort.15*, the filenames needed by *ADCIRC*. *ADCIRC* created the *fort.63* (global elevation) and the *fort.64* (global velocity) files. There are a couple of other files that hold basic output information, but we will only focus on the elevation and velocity files for the remainder of this tutorial.

When *ADCIRC* finishes running, click Exit.

10 Importing *ADCIRC* Global Output Files

Each output file from *ADCIRC* is imported into *SMS* as a “Dataset.” There are two types of datasets, scalar and vector. The global elevation file is an example of a scalar dataset, while the global velocity file is a vector dataset. You will import the global elevation file and the global velocity file simultaneously. To do this:

1. Select *File / Open*.
2. Hold the Shift key down and select both the *fort.63* and *fort.64* files.
3. Click the *Open* button.
4. Click *OK* in the *Convert to X MDF* dialog to convert both solution files.

SMS reads in the files and adds “Water Surface Elevation (63)” and “Velocity (64) mag” as *Scalar* datasets and “Velocity (64)” as a *Vector* dataset in the *Mesh Data* of the *Project Explorer*.


11 Viewing *ADCIRC* Output

Once both *ADCIRC* output files have been imported, the user must decide on how to view the data. The *Project Explorer* may be used to select the desired *Scalar* and *Vector* datasets.

- Activate the “Water Surface Elevation (63)” *Scalar* dataset by clicking on it in the *Project Explorer*.

11.1 Scalar Dataset Options

A good way to view the output is to edit the contour display options. To change the contour properties:

1. Select the Display Options  macro in the Toolbox.
2. In the 2D Mesh tab, click the *All off* button to turn off current display options.
3. Turn on the *Contours*, and *Mesh boundary*.
4. Under the Contours Options tab, change the *Contour Method* to Color Fill.
5. For the *Number of contours*, enter 25.
6. Click OK to exit the dialog box, and SMS will redraw the screen similar to Figure 6 (it will look about the same for the Time step 1 1:00:00).

To view the data at different time steps, select the desired time steps in the *Time Steps* window. The *Time Step* value is in hours, minutes and seconds from the start of the ADCIRC run.

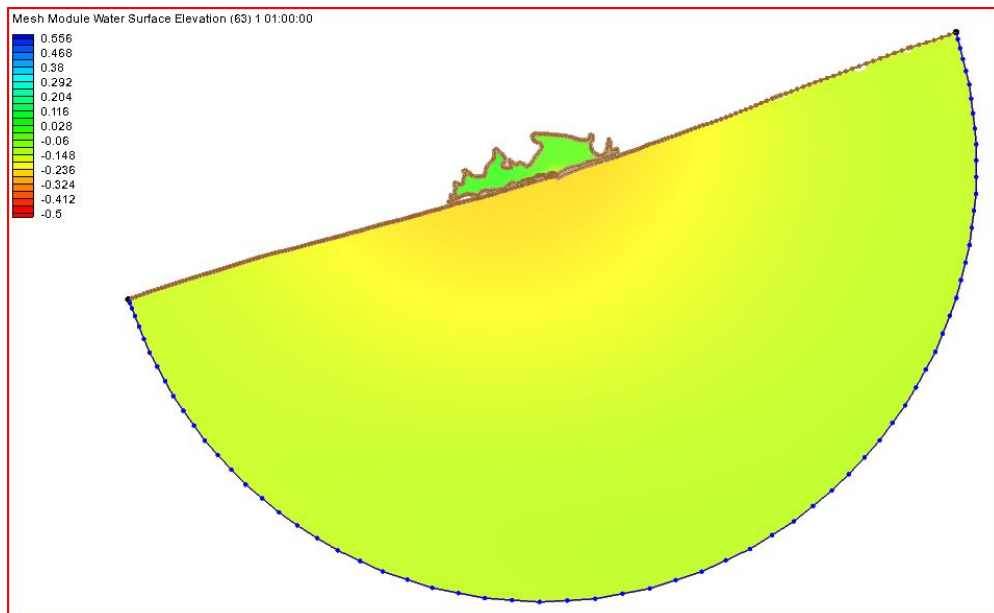




Figure 6 ADCIRC output from the “fort.63” file.

11.2 Vector Dataset Options

You can display velocity vectors several different ways. We will first view them displayed at each node, and then on a normalized grid.

Vectors at Each Node

1. Using the Zoom tool , zoom in on the mesh so only the bay area is visible.
2. Select the Display Options  macro in the Toolbox.
3. In the Display Options dialog under the 2D Mesh tab, turn on the *Vectors* toggle.
4. In the Vectors tab, under the Arrows Options section, make sure the *Shaft length* is set to Define min and max length.
5. Change the Min length to 15 and the Max length to 40.
6. Click the OK button to exit.

The screen should now look similar to that shown in Figure 7. You can now visualize the flow at each node through Shinnecock Bay at this particular time step.

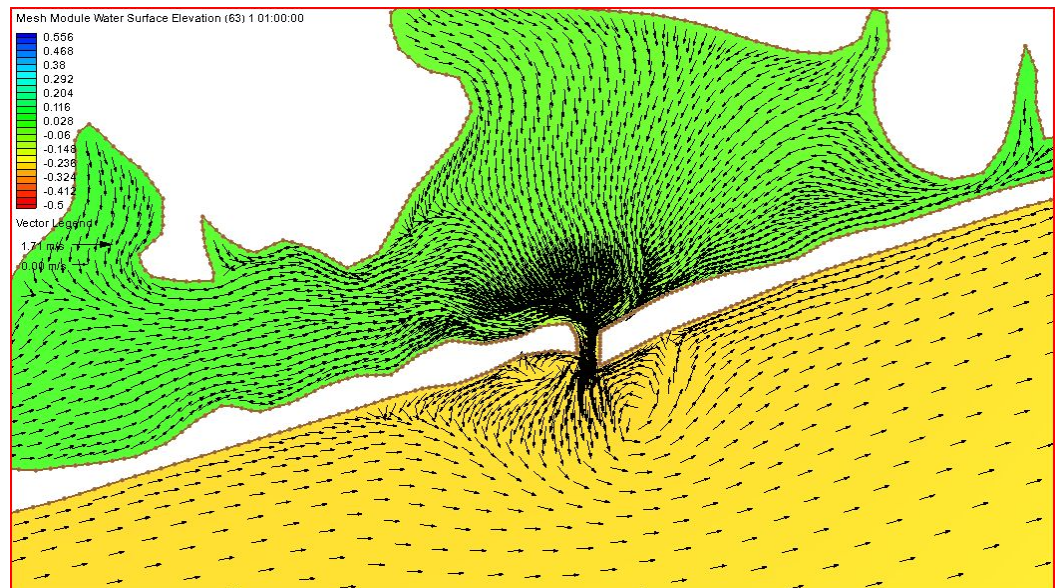



Figure 7 View of velocity vectors at each node.

Vectors on a Normalized Grid

1. Open the *Display Options*  and go to the *Vectors* tab.
2. Under Vector Display Placement and Filter, change the *Display* to the *on a grid* option.
3. For both the x pix and y pix, enter a value of 15 and click the OK button.

This method of displaying vectors is useful when displaying areas with both coarse and refined areas, such as the entire mesh in this case.

12 Film Loop Visualization

In addition to single time steps of contours and vectors, animations can be generated and saved. *SMS* enables the user to generate and save animations by using the Film Loop. To create a film loop of the ADCIRC analysis:

1. Select *Data | Film Loop*.
2. In the *Film Loop* dialog, click the *Next>* button.
3. Click the *Next>* button, then the *Finish* button.

SMS now starts the film loop, adding one frame at a time. Once the last frame has been added to the loop, an AVI Application will open and the animation will start automatically.

You may continue to experiment with the film loop features if you desire. Click the *Close* button when finished. The film loop has been saved as *sms.avi*.

13 Conclusion

This concludes the *ADCIRC Analysis* tutorial.